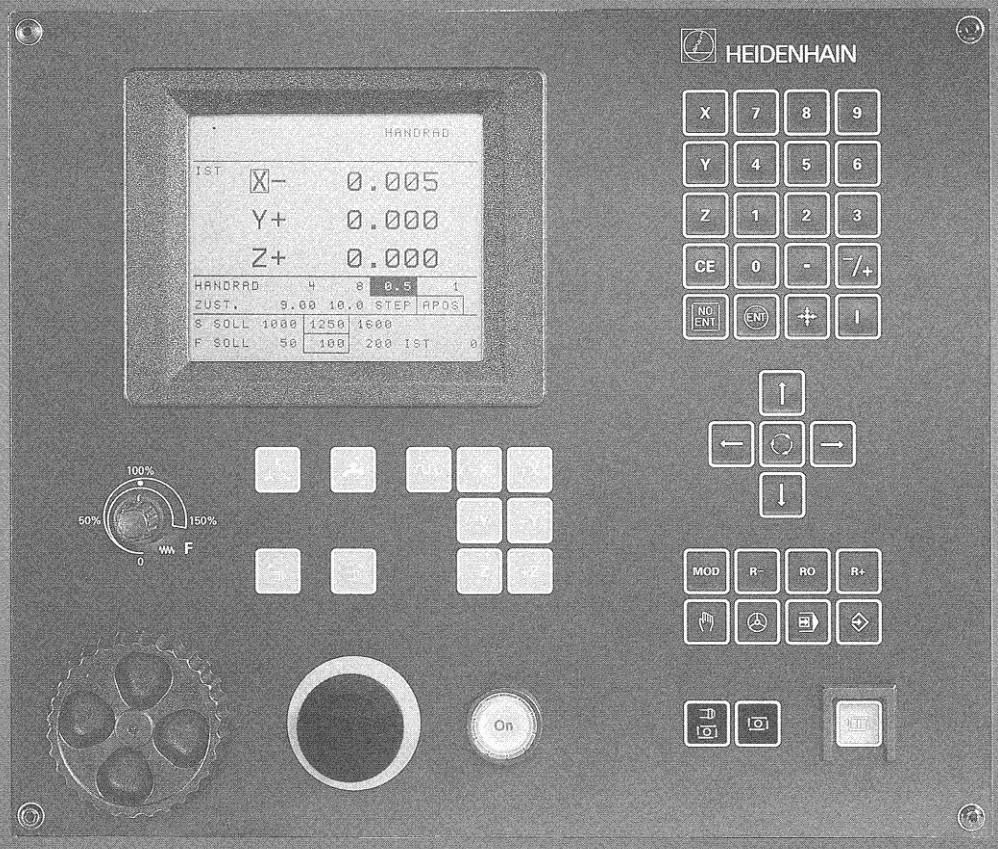


HEIDENHAIN

User's Manual

HEIDENHAIN Conversational Programming

TNC 116

January '93

Contents

General Information 3
INTRODUCTION 4
1 Short Description and Main Features 4
2 TNC 116 Operating Panel 5
3 The Keyboard 6
3.1 Keys for Operating and Programming the Control 6
3.2 Keys for Machine Functions 8
4 Operating Modes (Overview) 9
5 Operating Concept 10
BASICS 11
1 Part Programs 11
1.1 Cartesian Coordinate System 11
1.2 Right Hand Rule for Machine Axes 11
1.3 Datum Points 12
1.4 Absolute and Incremental Dimensions 12
1.5 Programming Tool Movement 13
1.6 Reference Marks 13
SWITCH-ON 14
1 Switch-On and Crossing the Reference Marks 14
MANUAL 15
1 MANUAL Operating Mode 15
1.1 Moving the Axes with the Axis Direction Keys 15
1.2 Selection of Feed Rate F and Spindle Speed S 16
1.3 Table: Feed Rate and Spindle Speed 16
2 HANDWHEEL Operating Mode 17
2.1 Moving the Axes with the HANDWHEEL 17
2.2 Jog Positioning 18
2.3 Moving to a Programmed Position (APOS) 19
3 Datum Setting 20
3.1 Datum Setting in the Infeed Axis 20
3.2 Datum Setting in the Working Plane 21
4 Tool Change 22
5 Switching on the Spindle 23
6 Turning Coolant On and Off 23
PROGRAMMING 24
1 PROGRAMMING Operating Mode 24
1.1 Editing 25
1.2 Transfer Actual Position to Program 28
PROGRAM RUN 29
1 PROGRAM RUN Operating Mode 29
1.1 Select Program/Block 29
1.2 Running a Program 30
1.3 Interrupting Program Run 30
1.4 Aborting Program Run 30
1.5 Making Changes to a Program 31
ADDITIONAL INFORMATION 32
1 MOD Functions 32
2 Parameters 32
2.1 Machine Parameters 32
2.2 User Parameters 32
2.3 Table: User Parameters 33
3 Error Messages 35
3.1 Table of Error Messages in Short Form 35
3.2 Clearing Error Messages from the Screen 36
3.3 Buffer Batteries 36

General Information

This User's Manual describes the complete scope of functions offered by the TNC 116. Since the machine builder has activated only those functions required for his machine, this handbook may describe functions that are not available on your TNC 116.

The software of the HEIDENHAIN TNC 116 undergoes continual development and improvement. For this reason, your software may deviate in some details from the description in this User's Manual.

Manufacturer's Certification

We hereby certify that the above unit is radioshielded in accordance with the German official register decree 1046/1984. The German postal authorities have been notified of the issuance of this unit and have been granted admission for examination of the series regarding compliance with the regulations.

If the unit is incorporated by the user into an installation then the complete installation must comply with the above requirements.

This manual describes the functions available on the TNC 116 starting with NC software number 246 111 01

INTRODUCTION

1 Short Description and Main Features

The HEIDENHAIN TNC 116 three-axis straight-cut control is a simple positioning control with program memory.

No special programming skills are needed to program the control. The necessary entries are requested by the control and displayed on the screen in plain language.

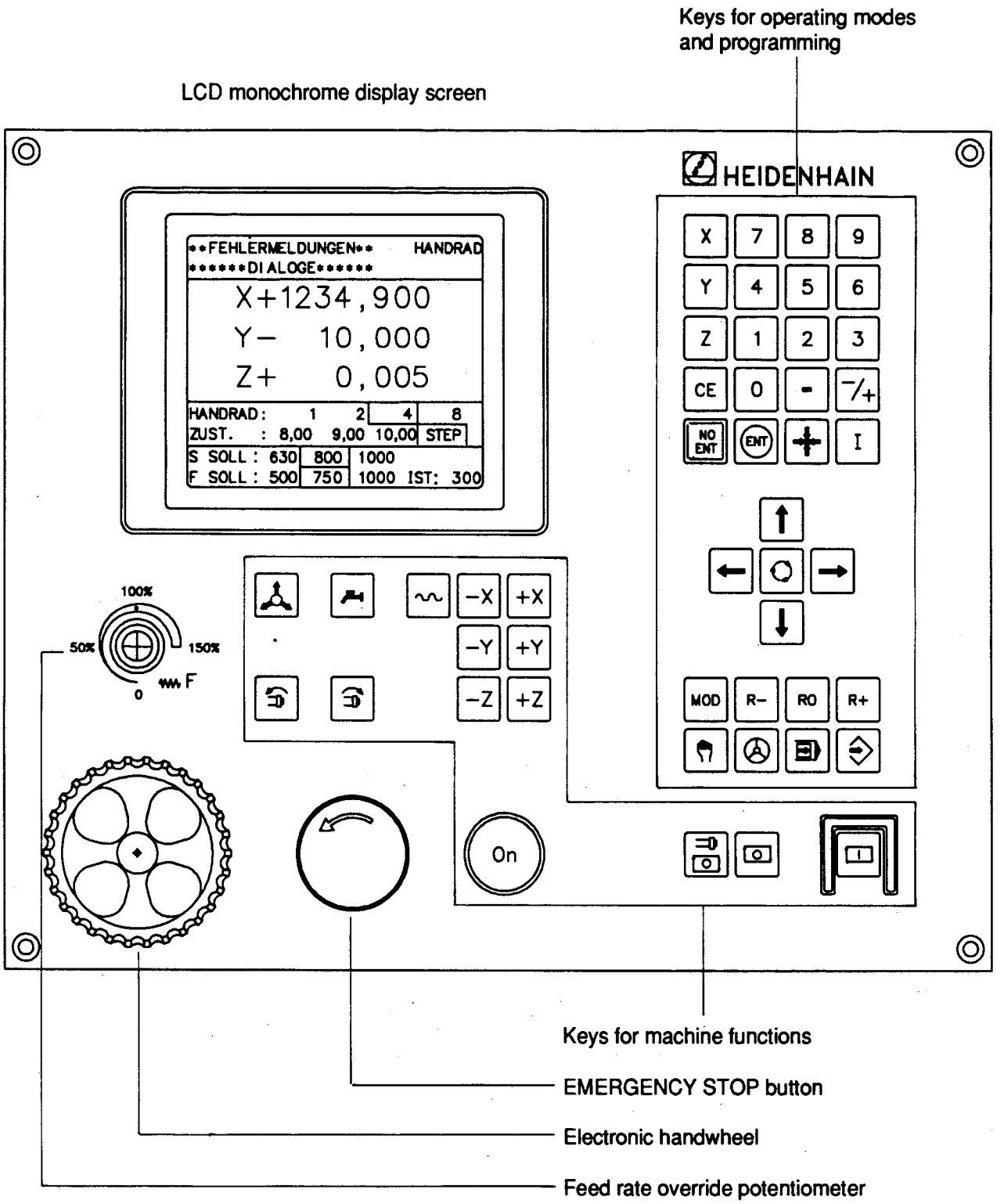
Programming errors, operating errors and malfunctions are recognized by the control, and the corresponding error messages are likewise displayed in plain language.

Main features of the TNC 116:

- LCD monochrome screen for display of actual position, status display and other possible entries
- A maximum of 500 positioning blocks can be stored and executed
- Positioning in metric or inch dimensions (selected with user parameters)
- Positioning in absolute and incremental dimensions
- Input of feed rate and spindle speed
- Feed rate adjustable during machining with override potentiometer
- Jog increments: each time an axis key is pressed the axis moves by the amount entered as the jog increment
- Input of tool radius and tool length compensation
- Electronic handwheel for fast movement of the axes during setup or machining
- Workpiece datum can be quickly located when the machine is powered up (e.g., after a power interruption)
- Keys for controlling the machine axes, spindle, coolant and tool holder

The HEIDENHAIN TNC 116 has been designed for workpiece machining on milling and boring machines. The operator programs the control directly at the operating panel.

2 TNC 116 Operating Panel

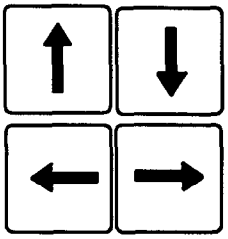
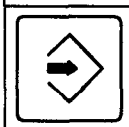
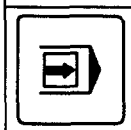
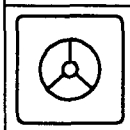


3 The Keyboard

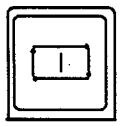
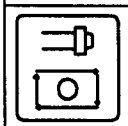
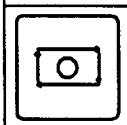
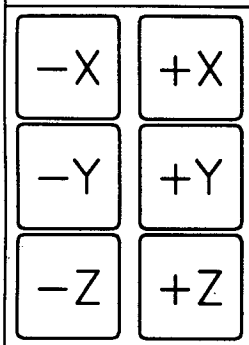
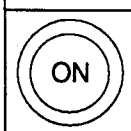
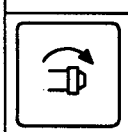
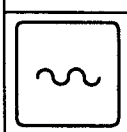
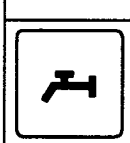
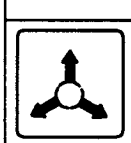
The keyboard contains one group of keys for operating and programming the control, and another group for machine functions.

3.1 Keys for Operating and Programming the Control

Keys	Functions
7 8 9 4 5 6 1 2 3 0	10-key numeric keypad
−/+	Plus/minus sign
.	Decimal point
X Y Z	Axis selection keys
CE	Clear entry, Clear error message
NO ENT	Skip dialog questions, Delete words
ENT	Confirm entry
	Capture actual position
I	Toggle between incremental and absolute dimensions

Keys	Functions
	Cursor keys
	SWITCH-OVER key for moving to different screen windows
R+	Radius compensation PLUS: lengthens the tool path by the value of the tool radius
RO	No radius compensation
R−	Radius compensation MINUS: shortens the tool path by the value of the tool radius
MOD	Additional operating modes/displays/settings
	PROGRAMMING operating mode
	PROGRAM RUN operating mode
	HANDWHEEL operating mode
	MANUAL operating mode

3.2 Keys for Machine Functions

Key(s)	Function
	START
	Feed / spindle STOP
	Feed STOP
	Axis direction keys
	Machine ON (luminous key)
	Spindle ON clockwise
	Spindle ON counterclockwise
	Positioning in rapid traverse Rapid traverse programming in PROGRAMMING operating mode
	Coolant ON/OFF
	Open/close tool holder

4 Operating Modes (Overview)

The following operating modes are available:

Manual

Axes move continuously as long as the axis keys are held down. Both feed and rapid traverse movements are possible.

Handwheel

For axis control with the electronic handwheel. Pressing the axis direction keys moves the axes by preselected increments, or to absolute positions.

Program Run

For stepwise or continuous movement to the positions in memory.

Programming

For entry of up to 500 axis positions in absolute or incremental dimensions, with or without tool radius compensation. Insertion and deletion of axis positions. Moving the axes with the axis direction keys and transfer of axis position values to memory.

Note: In all these operating modes you can preselect feed rate and spindle speed, and switch the spindle and coolant on or off.

5 Operating Concept

The TNC 116 displays various screen windows in which you enter values and program the control. To enter values, select the desired window with the switch-over key and then enter values on the numeric keypad, or use the cursor keys to choose values (such as the feed rate) from the range displayed.

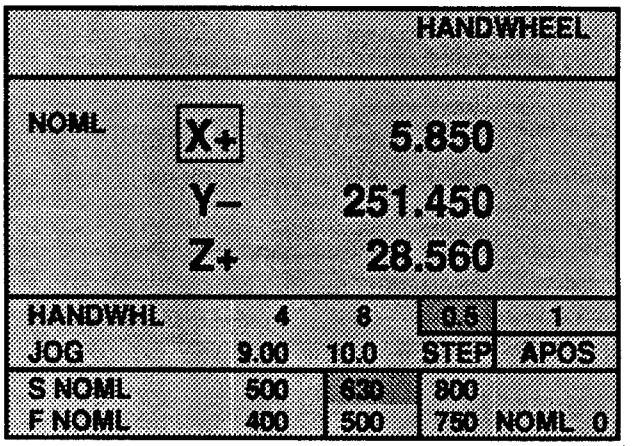
Example: Input of feed rate values for the HANDWHEEL operating mode

Active window

Actual position window

Handwheel and jog increment window

Spindle speed and feed rate window

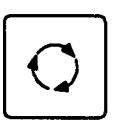


Select HANDWHEEL operating mode

The above screen is displayed.

▼

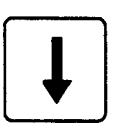
Select HANDWHEEL/JOG window



The selected window consists of two lines. The active line is indicated by an inverse window.

▼

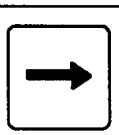
Select HANDWHEEL line



HANDWHL 4 8 [0.5] 1

▼

Select HANDWHEEL feed rate (e.g. 1)



HANDWHL 8 0.5 [1] 2

The inverted field indicates that a feed rate of 1 mm per handwheel revolution will be effective when the handwheel is turned.

This procedure is similar for all operating modes.

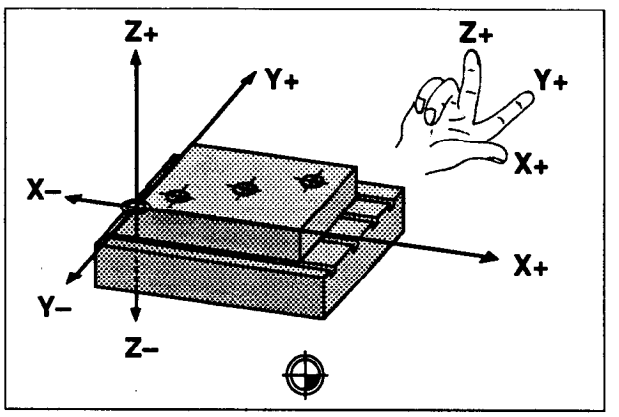
1 Part Programs

Workpiece machining procedures are defined in part programs. A part program contains such information as the target position towards which the tool is to move and the type of tool radius compensation (modifying the target position by taking the tool radius into account). Conversational programming makes it particularly easy to create part programs. The TNC 116 from HEIDENHAIN has been designed for the machinist who enters his programs into the control directly at the machine.

1.1 Cartesian Coordinate System

To define positions on the workpiece and over the traverse range of the machine, a coordinate system is required.

The Cartesian coordinate system consists of three imagined axes perpendicular to each other: the X-axis, the Y-axis and the Z-axis. The point of intersection of all three axes is called the *origin*. Think of the axes as rulers with millimeter or inch divisions. The positions on these axes are called the *coordinates*.



1.2 Right Hand Rule for Machine Axes

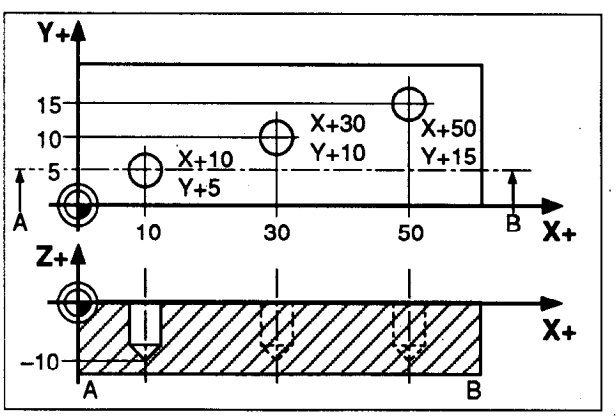
You can easily remember the traversing directions with the "right-hand rule." The positive direction of the X-axis is assigned to the thumb of the right hand, the Y-axis to the index finger and the Z-axis to the middle finger.

According to ISO 841 (DIN 66217), the Z-axis is defined by the direction of the tool axis. The positive Z direction points from the workpiece to the spindle. Axes X and Y are parallel to the remaining two machine slides.

Example

The illustration shows how positions on a workpiece are determined:

First, the origin of the coordinate system is placed at a suitable point on the workpiece. The center of the lower left hole is located 10 mm away from the origin on the X-axis and 5 mm away from the origin on the Y-axis. The coordinates of the hole center are therefore X+10 and Y+5. The holes are 10 mm deep, so their Z coordinate is Z–10.



1.3 Datum Points

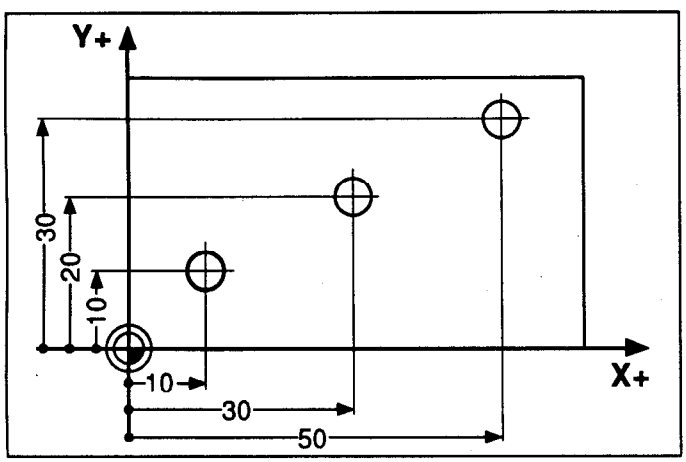
Before you start machining, clamp the workpiece on the machine table and align it so that the axes of the imagined coordinate system are parallel to the machine axes.

The control now needs to know the coordinates of a known position, for example the origin of the coordinate system on the workpiece. This is called the *workpiece datum*. The control uses the workpiece datum as a reference point for all other positions on the workpiece. To set the datum, conventional aids such as optical contour finders, centering devices or edge finders can be used.

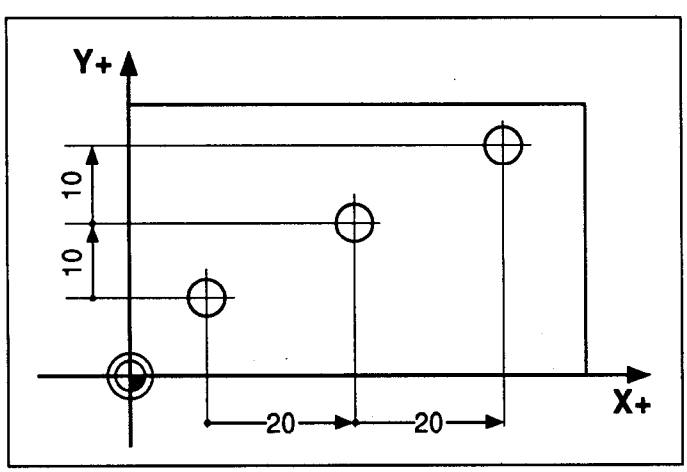
1.4 Absolute and Incremental Dimensions

There are two types of dimensions in Cartesian coordinates:

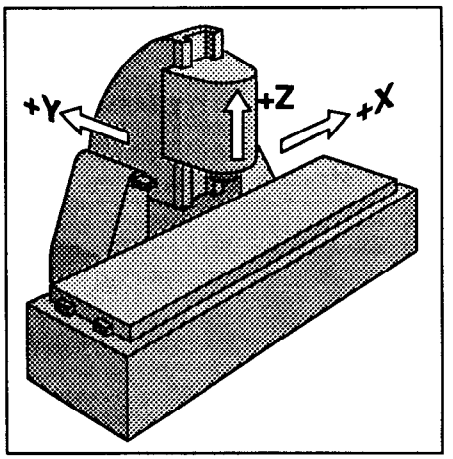
- Absolute dimensions. The coordinates you enter are based on the current datum. The machine moves ***to*** a certain position.



- Incremental dimensions. The coordinates you enter are based on the last nominal position of the tool. The machine moves ***by*** a certain distance.



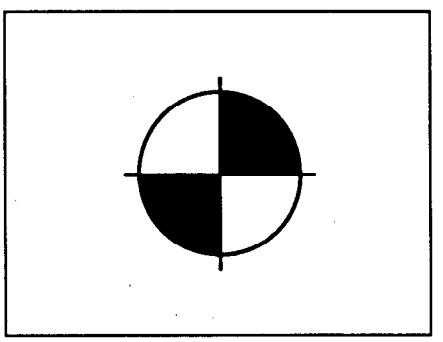
1.5 Programming Tool Movement



During machining, either the machine table with the clamped workpiece moves or the tool itself moves.

However, you always program as if the tool moves and the workpiece remains stationary.

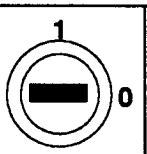
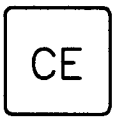
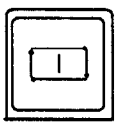
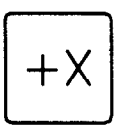
1.6 Reference Marks



The linear encoders on the machine have one or more reference marks. A reference mark identifies a certain axis slide position (indicated on the machine with the symbol to the right). After a power interruption, the reference marks of all axes must be crossed over so that the TNC can reestablish the relationship between display values and machine slide positions. The reference marks generate a signal from which the TNC can identify that position as a reference mark.

If the linear encoders have **distance-coded reference marks,** the machine axes only have to be moved by a maximum of 20 mm after a power interruption.

1 Switch-On and Crossing the Reference Marks

	Switch on the power supply.
▼	
MEMORY TEST POWER INTERRUPTED 	The control checks the internal program memory. Press CE to clear the message. The control then checks the functioning of the EMERGENCY STOP circuit.
▼	
RELAY EXT. DC VOLTAGE MISSING	Switch on the control voltage. The ON button must light up.
▼	
TRAVERSE REFERENCE POINTS X-AXIS Y-AXIS Z-AXIS  	Move the axes in the indicated sequence (repeat the procedure for each axis). or Move the axes with the axis direction keys. The sequence of axes is defined by the machine builder.
▼	
MANUAL	The MANUAL operating mode is automatically selected.

Note: If different sequence is required, for example to avoid collisions with the workpiece, change the sequence in machine parameter MP 1340.

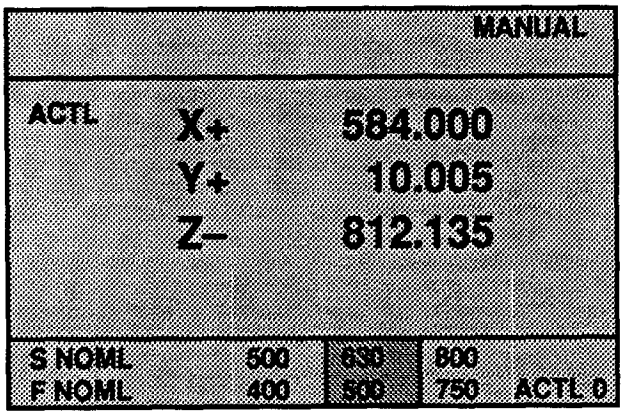
It is also possible to cross over the reference marks in any sequence with the axis direction keys.

1 MANUAL Operating Mode

Operating mode, error message
Dialog line

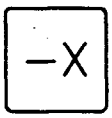
Position display window

Spindle speed
Feed rate



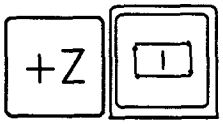
1.1 Moving the Axes with the Axis Direction Keys

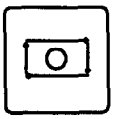
To move the axes:

MANUAL	
	Press the direction key for the desired axis. The axis keeps moving as long as the key is held down.

More than one axis can be moved at the same time in this way.

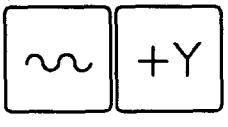
To move the axes continuously:

MANUAL	
 SIMULTANEOUSLY	Press and hold down the direction key for the desired axis and press the START key. The axis continues to move after the keys are released.

	To stop the axis: press the STOP key.

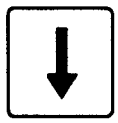
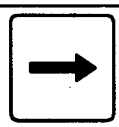
It is only possible to move one axis at a time in this way.

To move the axes at rapid traverse speed:

MANUAL	
 SIMULTANEOUSLY	Press and hold down the axis direction key for the desired axis and press the rapid traverse key. The axis moves at rapid traverse (using the feed rate from machine parameter)

1.2 Selection of Feed Rate F and Spindle Speed S

Select desired operating mode

MANUAL, HANDWHEEL, PROGRAMMING OR PROGRAM RUN	
	Use the switch-over key to select the window for spindle speed and feed rate
	With the vertical cursor keys, select the line (such as spindle speed S NOML)
	With the horizontal cursor keys, select the desired value (see table)

1.3 Table: Feed Rate and Spindle Speed

Spindle Speed S [RPM]	Feed Rate F [mm/min]
20	50
31.5	100
40	200
50	300
63	400
80	500
100	750
125	1000
160	2500
200	5000
250	
315	
400	
500	
630	
800	
1000	
1250	
1600	
2000	
2500	
3150	
4000	

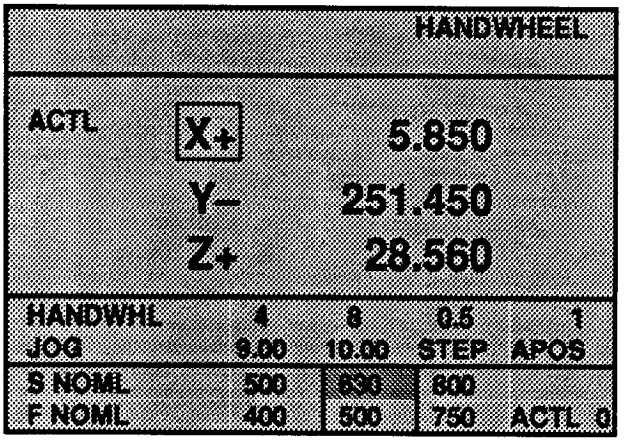
Note: The feed rate can be varied with the override control from 0–150% of the programmed rate. The gear range for the spindle speed is changed manually or automatically, depending on the machine.

2 HANDWHEEL Operating Mode

Operating mode, error message
Dialog line
Window for active axis

Position display

Traverse in mm per handwheel rev.
Increment for axis direction keys
Spindle speed
Feed rate



2.1 Moving the Axes with the HANDWHEEL

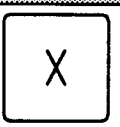
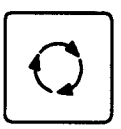
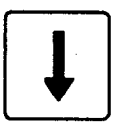
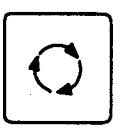
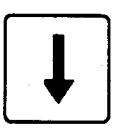
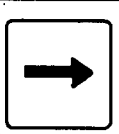
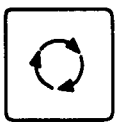
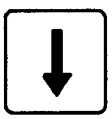
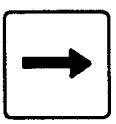
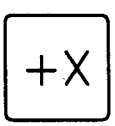
HANDWHEEL	
e.g.  or  	Select the desired axis with the axis selection keys or use the SWITCH-OVER key and the vertical CURSOR KEYS. The inverse display indicates the active axis.
	Select the window HANDWHL/JOG with the SWITCH-OVER key.
	Select the HANDWHEEL line with the vertical CURSOR KEYS.
	With horizontal CURSOR KEYS, select the HANDWHEEL feed rate. You can then move the selected axis by turning the handwheel.

Table: HANDWHEEL Feed Rate

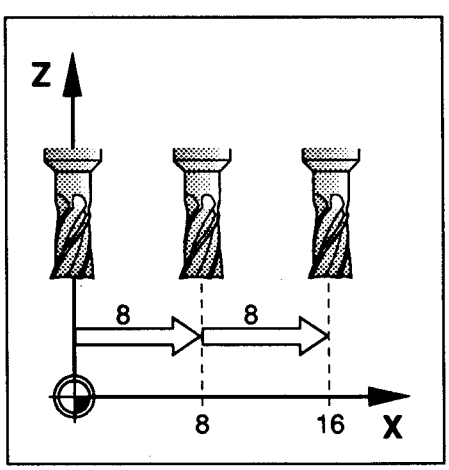
HANDWHEEL Feed Rate [mm/rev.]					
0.5	1	2	4	8	

2.2 Jog Positioning

With jog positioning, the machine axis moves by the selected amount each time the axis direction key is pressed.

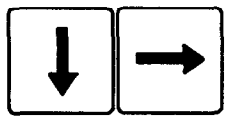
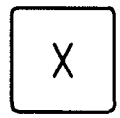
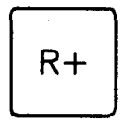
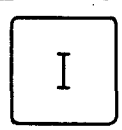
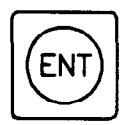
HANDWHEEL	
	Select the window HANDWHL/JOG/APOS with the SWITCH-OVER key.
	Select the JOG line with the vertical CURSOR KEYS.
	With the horizontal CURSOR KEYS, select the jog increment (0.5, 1, 2, 4 or 8). or
STEP	Select "STEP". The dialog line displays the question "JOG INCREMENT?". Key in a value with the numeric keypad and confirm with ENT.
e.g. 	You can now move any axis by the selected jog increment by pressing the axis direction key for that axis.

Note: You can press the axis direction keys repeatedly as many times as you want; the axis moves by the amount of the jog increment each time.

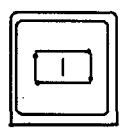


2.3 Moving to a Programmed Position (APOS)

The axis programmed in the positioning block moves to the programmed position. The tool radius is taken into account.

HANDWHEEL	
	Select the window HANDWHL/JOG/APOS with the SWITCH-OVER key.
	Select the dialog APOS with the CURSOR KEYS. The question "POS. NOM. VALUE?" appears in the dialog line.
	Press the axis selection key for the desired axis.
	Enter the axis position with the numeric keypad.
	Press the key for radius compensation R–, R0 or R+.
	Press the INCREMENTAL/ABSOLUTE key if positioning is to be in incremental dimensions.
	Press ENT to transfer the program block to memory.

Before executing the program block, turn on coolant and spindle if necessary.

	Press the START key. Execution of the program block is started.

Note: To interrupt program run, see the chapter PROGRAM RUN.

3 Datum Setting

The datum setting procedure gives the TNC the coordinates of a known workpiece position.

Preparation

Clamp and align the workpiece.

▼

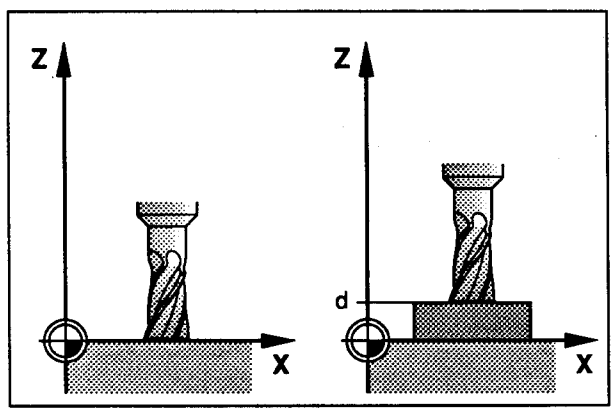
Insert zero tool with known radius.

▼

	Select MANUAL operating mode.

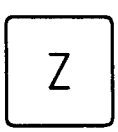
3.1 Datum Setting in the Infeed Axis

Protective measures:
If the workpiece surface should not be scratched by the tool, place a sheet of metal of known thickness *d* on the workpiece. The value that you then enter for the datum in the infeed axis must be increased by the value of *d*.

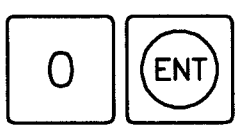


Move the tool until it touches the workpiece surface.

▼

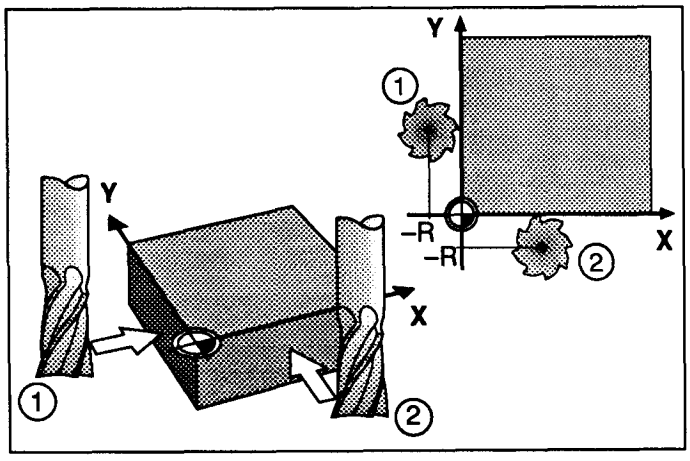
e.g. 	Select the infeed axis.

▼

e.g. 	Zero tool: set datum to Z = 0 or enter thickness *d* of the metal sheet.

Note: If you open the dialog for datum setting by mistake (you don't want to set a datum point), just press NO ENT or one of the operating mode keys.

3.2 Datum Setting in the Working Plane



Touch a workpiece edge with the zero tool:	
e.g. [X]	Select the axis.
e.g. [5] [−/+] [ENT]	Enter the position of the tool center including the algebraic sign (example: X = –5 mm) for the selected axis.

Repeat this procedure for all axes in the working plane.

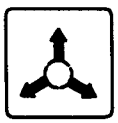
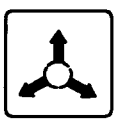
Note: For each tool, the TNC takes into account the compensation values for tool length in the spindle axis and for the tool radius in the working plane. The corresponding entries can be made in the PROGRAMMING or PROGRAM RUN operating modes.

4 Tool Change

The tool can be changed in any operating mode.

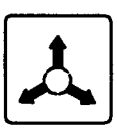
Prerequisite: The spindle must be stopped.

To insert the tool into the spindle:

	Switch off the spindle.
	Open the tool holder (if closed) and insert the tool.
	Clamp the tool.

To remove the tool from the spindle:

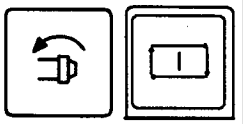
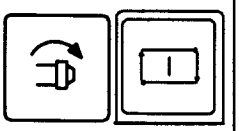
Prerequisite: The spindle must be stopped.

	Switch off the spindle.
	Press the tool holder key and at the same time remove the tool from the spindle.

Caution: Keep your hands out of the machining area while the spindle is turning.

5 Switching on the Spindle

The spindle can be switched on if there is a tool in the spindle.

	**Spindle on counterclockwise:** Press the key for SPINDLE ON COUNTERCLOCKWISE, then press the START key.
	**Spindle on clockwise:** Press the key for SPINDLE ON CLOCKWISE, then press the START key.

6 Turning Coolant On and Off

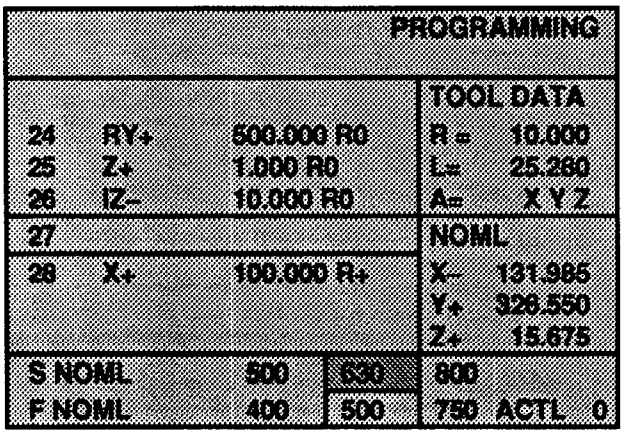
	To turn the coolant on or off, press the coolant key.

1 PROGRAMMING Operating Mode

Operating mode, error message
Dialog line
Tool data
Prog. block with rapid traverse

Empty block
Program block

Spindle speed
Feed rate
ACTUAL position



Programs are entered in the PROGRAMMING operating mode.

In this operating mode, you can enter programs, check them, or change them. The program is not run.

A maximum of **500 program blocks** (axis positions) can be stored in the **program memory**.

A **part program** consists of individual program blocks. If different programs are to be stored, the **program end** should be indicated by entering an **empty block** with the NO ENT key (see block number 27 in illustration above).

To run a program, select the PROGRAM RUN operating mode.

Block number — The sequential block numbers identify individual program blocks within the part program.

Block — Each block in the program represents one work step.
Example: Y+500 R0

Word — A word consists of individual commands.
Example: Y+500

Address
Value — Words are further divided into address letters (such as "Y") and values (such as "500").

Empty block — An empty block is entered by pressing the NO ENT key. An empty block identifies the end of the program.

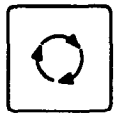
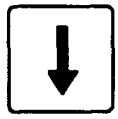
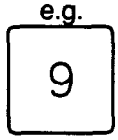
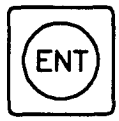
Note: In the PROGRAMMING operating mode, the feed rate (F) and the spindle speed (S) can be selected and the tool data (L, R) edited. In addition, the spindle and the coolant can be switched on and off with the appropriate keys.

The axes can be moved with the axis direction keys in the MANUAL operating mode.

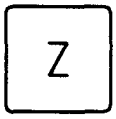
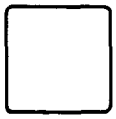
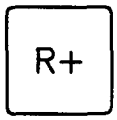
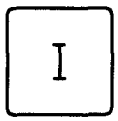
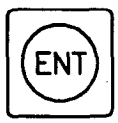
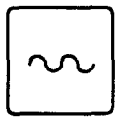
1.1 Editing

"Editing" refers to entering, changing, adding to, deleting and checking programs. The editing functions assist the operator in selecting and changing program blocks and words.

To select a block:

EDITING	
	Select the window with the program blocks by pressing the SWITCH-OVER key.
 or e.g. 	Select the block number with the CURSOR KEYS. or Enter the block number with the NUMERIC KEYPAD. In the dialog line, the message "GOTO: NUMBER" appears.
	Then press the ENTER key to select the block.

To enter a block:

EDITING	
	Press axis selection key X, Y or Z. The dialog line displays the prompt "POS. VALUE?"
	Enter the axis position with the NUMERIC KEYPAD.
	Select the radius compensation: R–, R0 or R+.
	If you want to position in incremental dimensions, press the INCREMENTAL/ABSOLUTE key.
	Press ENT to transfer the program block to memory or
	Press and hold down the RAPID TRAVERSE key, and press the ENT key. The program block is transferred to memory with an R (for "Rapid") before the axis designation.

To insert a block:

➤ New blocks can be inserted at any point in a program. First call up the block after which you want to insert the new block (see "To select/enter a block" above). The control automatically renumbers the following blocks.

To change a block:

➤ Existing blocks in a program can be changed at any time. Just select the block you want to change (see "To select a block" above).

➤ The block can now be changed. Confirm your changes with the ENTER key.

To delete a block:

EDITING	
	Select the block you want to delete (see "To select a block" above).
CE	Press the CE key. The dialog line displays the dialog question CLEAR=ENT/END=NOENT
ENT	Press ENT to delete the program block. The block numbers of the following blocks are automatically renumbered.

Deleting the block memory

To delete the entire block memory:

MANUAL	
MOD	In the MANUAL operating mode, press MOD. The dialog line displays the dialog question SELECT=ENT/END=NOENT and the following menu appears on the screen: CLEAR BLOCK MEMORY CLEAR VACANT BLOCKS EDIT USER PARAMETERS ENTER CODE NUMBER
↓	With the cursor, select "CLEAR BLOCK MEMORY".
ENT	The dialog line displays the dialog question "CLEAR=ENT/END=NOENT".
ENT	Press ENT to delete the block memory.

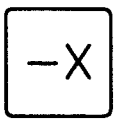
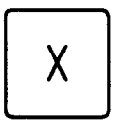
Deleting empty blocks

Empty blocks in the block memory separate the individual programs from each other.
To delete all empty blocks:

MANUAL	
MOD	The dialog line displays the dialog question SELECT=ENT/END=NOENT and the following menu appears on the screen: CLEAR BLOCK MEMORY CLEAR VACANT BLOCKS EDIT USER PARAMETERS ENTER CODE NUMBER
↓	With the cursor, select "CLEAR VACANT BLOCKS".
ENT	The dialog line displays the dialog question "CLEAR=ENT/END=NOENT".
ENT	Press ENT to delete the empty blocks.

1.2 Transfer Actual Position to Program

The TNC 116 has the capability of transferring an actual axis position to block memory. This feature allows you to generate a program while machining a workpiece in the MANUAL operating mode. You can then use the program so generated to machine an identical workpiece.

PROGRAMMING	
	With the axis direction keys (–X, +X, –Y etc.), move to the position you want to transfer.
	Press the desired axis selection key (X, Y or Z).
	Press the CAPTURE ACTUAL POSITION key to transfer the axis position to memory.

1 PROGRAM RUN Operating Mode

In the operating mode PROGRAM RUN, the control executes the program in memory. The program is executed blockwise — you must restart the program after each block.

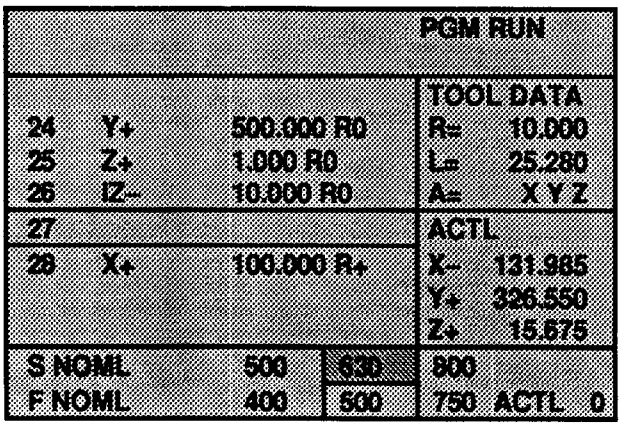
Operating mode, error message
Dialog line
Tool data

Input of tool axis

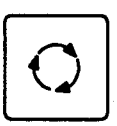
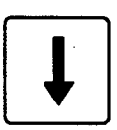
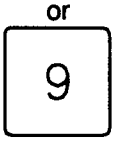
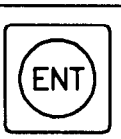
Empty block
Program block

Spindle speed
Feed rate

Actual position

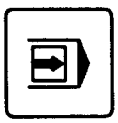


1.1 Select Program/Block

PROGRAM RUN	
	Use the SWITCH-OVER key to select the window with the program blocks.
 or 	Select the block number with the CURSOR KEYS. Or Key in the block number on the NUMERIC KEYPAD. The dialog line displays "GOTO: NUMBER ="
	Press ENTER to select the block number you keyed in. Or Press and hold down the PROGRAM RUN key and press the START key. The program is run block by block. Program run stops and the program ends when an empty block is reached.

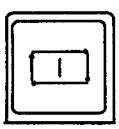
1.2 Running a Program

When you have selected a program, it can be executed blockwise:



- Select PROGRAM RUN operating mode

- Go to the beginning of the program
- Select feed rate F and spindle speed S
- Enter the tool data
- Insert the tool
- Switch on the spindle
- Switch on the coolant

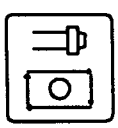


- Press the START key. The program block is executed.
  After a block has been executed, you must press the START key again to execute the next block. **The program ends when an empty block is reached.**

Note: The selected feed rate can be adjusted with the feed rate override control.

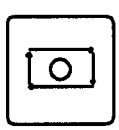
1.3 Interrupting Program Run

You can halt the program while a program block is being executed:



- Press the feed / spindle stop key

or



- Press the feed stop key.

Program run stops. To finish the interrupted program block, press the START key again.

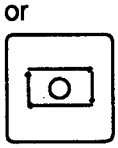
Note: When the execution of a program block is interrupted, the symbol "*" blinks in the actual position window.

1.4 Aborting Program Run

A program can be aborted while a block is being executed:

- Press the feed rate/spindle stop key **twice**.

or



- Press the feed stop key **twice**.

1.5 Making Changes to a Program

You can edit a program after you have aborted program run. To edit the program you must switch to the PROGRAMMING operating mode. The program block can now be edited (see the chapter "PROGRAMMING").

EMERGENCY STOP

In case of danger, the machine can be shut down by pressing the EMERGENCY STOP button.

To resume working, reset the EMERGENCY STOP button by turning it clockwise. If the power was interrupted, you must cross over the reference marks.

Then:

- ➤ Locate and correct the error
- ➤ Switch on the control voltage
- ➤ Clear the screen with the CE key
- ➤ Restart program run

ADDITIONAL INFORMATION

1 MOD Functions

Besides the operating modes there are also the MOD ("mode") functions. These functions allow additional displays and settings.

To select the MOD functions:

MANUAL	
MOD	Press the MOD key. The dialog line displays the dialog query SELECT=ENT/END=NOENT. The following menu appears in the lower half of the screen: CLEAR BLOCK MEMORY CLEAR VACANT BLOCKS EDIT USER PARAMETERS ENTER CODE NUMBER
↓	Select the desired function with the vertical CURSOR KEYS.
ENT	Confirm your selection by pressing ENT. **Then follow the instructions in the dialog line.**
	Make your entries according to the selected function.

2 Parameters

The TNC 116 contains two types of parameters: machine parameters and user parameters.

2.1 Machine Parameters

Machine parameters are entered into the parameter memory by the machine builder and do not normally need to be changed by the user.

2.2 User Parameters

User parameters are parameters that often need to be entered or changed by the machine operator. To access the user parameters, select operating mode MANUAL and press the MOD key (see MOD function).

2.3 Table: User Parameters

User Parameters	Function and entry	Reaction	
MP1340.0–2	Axis sequence when crossing the reference marks. Entry range: 0 to 3 0 = No reference mark evaluation 1 = X axis first 2 = Y axis first 3 = Z axis first		
MP4310.0	*Function defined by machine builder*		
MP4310.1	*Function defined by machine builder*		
MP7210	Programming station Entry values: 0, 1, 2 0 = Control and programming 1 = Programming station "PLC active" 2 = Programming station "PLC not active"	RESET	
MP7230	Dialog language Entry range: 0 to 4 0 = German 1 = English 2 = French 3 = Italian 4 = Spanish		
MP7285	Calculation of tool length in the position display of the tool axis Entry values: 0 or 1 0 = Tool length ignored 1 = Tool length taken into account		
MP7320	Contrast for LCD monitor screen Entry range: 0 to 15 0 = Lowest contrast 15 = Highest contrast		
MP7321	Unit of measurement: mm or inches Entry values: 0 or 1 0 = mm 1 = inch		
MP7322	Switch-over of the position display window Entry range: 0 to 2 0 = ACTUAL position (ACTL) 1 = Servo lag (LAG) 2 = Reference position (REF)		

User parameters	Function and entry	Reaction	
MP7323	Enable "Program run full sequence" Entry values: 0 or 1 0 = No "Program run full sequence" 1 = "Program run full sequence" (hold down PROGRAM RUN key and press NC START key)		
MP7680	Memory function for axis direction keys Entry values: 0 or 1 0 = not stored 1 = stored		

Note: To leave the machine parameter list, press any operating mode key.

3 Error Messages

The TNC 116 features an automatic error diagnostics system. Faults and malfunctions recognized by the control are displayed in the first line of the LCD monitor in plain language.

3.1 Table of Error Messages in Short Form

Error messages which can be cleared with CE

ERR 1	"POWER INTERRUPTED"
ERR 2	"KEY NON-FUNCTIONAL"
ERR 3	"ENTRY VALUE INCORRECT"
ERR 7	"COMPENSATION VALUES DELETED"
ERR 9	"EMERGENCY STOP"
ERR 14	"PROGRAM MEMORY EXCEEDED"
ERR 20	"NC: PROGRAM MEMORY ERASED"
ERR 27	"OPERATION PARAMETERS ERASED"
ERR 31	"RADIUS COMP. UNDEFINED"
ERR 32	"ERRONEOUS LIMIT SWITCH VALUE X"
ERR 33	"ERRONEOUS LIMIT SWITCH VALUE Y"
ERR 34	"ERRONEOUS LIMIT SWITCH VALUE Z"
ERR 40	"EXCHANGE BUFFER BATTERY"
ERR 63	"NC BLOCK NOT COMPLETE"
ERR 70	"LIMIT SWITCH X+
ERR 71	"LIMIT SWITCH X–
ERR 72	"LIMIT SWITCH Y+"
ERR 73	"LIMIT SWITCH Y–"
ERR 74	"LIMIT SWITCH Z+"
ERR 75	"LIMIT SWITCH Z–"
ERR 100	"MACHINE ERROR"
ERR 101	"OIL TANK EMPTY"
ERR 102	"ACKNOWLEDGMENT 0-SIGNAL"
ERR 103	"ACKNOWLEDGMENT 1-SIGNAL"
ERR 104	"SPINDLE ACCELERATION ERROR"
ERR 105	"SPINDLE BRAKING ERROR"
ERR 106	"UNDEF. GEAR POSITION"
ERR 107	"HANDWHEELS ONLY IN MANUAL MODE"

Error messages which cannot be cleared

FERR 1	"PROCESSOR CHECK ERROR A"
FERR 3	"EMERGENCY STOP DEFECTIVE"
FERR 4	"TNC-OPERATING TEMP. EXCEEDED"
FERR 5	"MEASURING SYSTEM X DEFECTIVE A"
FERR 6	"MEASURING SYSTEM Y DEFECTIVE A"
FERR 7	"MEASURING SYSTEM Z DEFECTIVE A"
FERR 8	"PROCESSOR CHECK ERROR 4"
FERR 9	"PROCESSOR CHECK ERROR 4"
FERR 10	"PROCESSOR CHECK ERROR 6"
FERR 11	"PROCESSOR CHECK ERROR B"
FERR 12	"PROCESSOR CHECK ERROR B"
FERR 13	"PROCESSOR CHECK ERROR B"
FERR 14	"PROCESSOR CHECK ERROR B"
FERR 15	"GROSS POSITIONING ERROR X"
FERR 16	"GROSS POSITIONING ERROR Y"
FERR 17	"GROSS POSITIONING ERROR Z"

3.2 Clearing Error Messages from the Screen

When an error message appears in the first line of the screen, the control is inhibited. You can only resume operation when the error message has been acknowledged and cleared.

Non-blinking error messages can be cleared by pressing the **CE** key.

Blinking error messages can only be cleared by **switching off the main switch** of the machine.

Blinking error messages signal serious machine malfunctions and always entail an automatic EMERGENCY STOP.

When the machine and control have been switched off, the error must be corrected.

Note: If error messages reappear when power is switched on again, contact the machine builder's customer service department.

See also EMERGENCY STOP on page 31.

3.3 Buffer Batteries

Programs and machine parameters are stored in non-volatile memory — they are not erased when the power is switched off.

If the message "EXCHANGE BUFFER BATTERY" appears in the display, change the batteries.

The batteries should be changed at least once a year.

Battery type:
Three AA-size, leak-proof
IEC designation: LR6

Important Note:

Do not turn off the main switch on the switch cabinet when you are changing the buffer batteries. Otherwise the program memory contents and the parameter settings will be lost when the batteries are removed.

The buffer batteries are located in a removable battery holder on the left side of the control. When changing the batteries, be sure to note the correct orientation (symbols for +/– poles shown on battery holder).

If the batteries lose their charge completely or are removed, the program memory will continue to be supplied with power as long as the main switch of the machine stays on. The control remains ready for operation. However, the program memory will be erased if the main switch is accidentally turned off. If this happens, you must:

- Check the machine parameters, and if necessary re-enter them
- Reset the workpiece datum

HEIDENHAIN

281 385 20 · 2 · SW01 · 1/93 · Printed in Germany · Subject to alteration

Paper bleached without chlorine!